ABSTRACT
In this paper the development of a system simulation model that can predict the dynamic behaviour of the Pebble Bed Modular Reactor (PBMR) will be described. The model predicts the transient mass, momentum and energy transfer in all components in a coupled manner. Very important is that the nuclear reactor and heat exchangers are not treated as lumped systems but as 2-D distributed systems. This allows one to take full account of the thermal inertia of the system. Another feature of the model is that it can deal with conduction heat transfer through solid structures.

The paper will focus on the overall modelling approach and the simultaneous solution of the mass, momentum and energy equations.

A few cases emanating from a benchmark study are presented in this paper. This includes the sudden closure of a valve in a pipeline, transient heat transfer through a reservoir wall and transient heat transfer in the recuperator.

INTRODUCTION
The Pebble Bed Modular Reactor (PBMR) is a new type of high temperature gas nuclear reactor currently being developed by the company PBMR (Pty.) Ltd. in South Africa. The heat released in the nuclear reactor is converted to work in a recuperative Brayton Cycle.

When designing a plant such as the PBMR one must not only be able to predict the steady-state performance of the system as a whole (including all the heat transfer equipment) but also the dynamic behaviour of the system. This is necessary for the design of the operating procedures and control system.

Systems simulations of complete systems such a power plants present unique challenges in dealing with issues such as speed, stability, accuracy, flexibility and the interfacing of different types of components such as turbines, compressors, valves and heat exchangers.

With simulating complete systems the major objective is not to model the processes in each component in great detail but to accurately predict the performance of the system as a whole. However, with the modelling of sub-systems one must avoid over simplifications that may negatively impact the accuracy of the overall system simulation.

In this paper the development of a system simulation model, which can predict both the steady state and the dynamic behaviour of the PBMR, is discussed. The model predicts the transient mass, momentum and energy transfer in all components in a coupled manner. Very important is that the nuclear reactor and heat exchangers are not treated as lumped systems but as distributed systems. This allows one to take full account of the thermal inertia of the system. Another feature of the model is that it can deal with conduction heat transfer through solid structures.

NOMENCLATURE
\( d \) mass source
\( E \) energy transfer to node or element i.e. heat transfer to element minus shaft work
\( f \) frictional pressure drop as function of \( \rho \) and \( Q \)
\( g \) gravitational acceleration
\( h \) enthalpy
\( m \) mass
\( \dot{m} \) mass flow rate
\( p \) pressure
\( Q \) volumetric flow rate
\( r \) continuity error
\( s \) value which indicates whether the positive flow direction of an element is directed toward a node or away from the node with \( s = 1 \) towards the node and \( s = -1 \) away from the node
\( t \) time
\( T \) temperature
\( V \) volume
\( x \) length
\( z \) elevation

Greek Letters
\( \alpha \) time integration weighing factor
\( \rho \) density
\( \Delta \) increment

Subscripts
\( amb \) ambient condition
\( i \) node
NETWORK APPROACH TO MODELLING OF THERMAL-FLUID SYSTEMS

The approach that will be followed is a general network approach where components such as compressors and heat exchangers are represented by one-dimensional elements. Elements, denoted by circles, can be connected at their end points to form any arbitrary network as shown in Figure 1. The end points of elements are called nodes and are denoted by squares.

![Figure 1: Arbitrary network created by connecting elements at common nodes.](image1)

An important feature of the model is that elements such as pipes, heat exchangers and the reactor, although depicted on the systems level as single elements or pairs of elements, can be discretized into sub-networks. Networks can therefore be embedded within networks thereby enabling the model to treat complex elements as distributed systems rather than lumped systems.

Consider for instance the recuperator depicted on the systems level as two associated elements as shown in Figure 2.

![Figure 2: Recuperator depicted as a pair of associated elements on the systems level.](image2)

Internally the recuperator is discretized as shown in Figure 3.

![Figure 3: Discretization of the recuperator.](image3)

In Figure 3 the smaller circles denote convective heat transfer links while the smaller squares denote temperatures nodes in the metal separating the hot and cold streams.

The network approach allows for the discretization of more complex components such as shell and tube heat exchangers. Figure 4 shows the discretization of part of a shell and tube heat exchanger. As in the previous figure larger circles and squares denote flow elements and nodes respectively while smaller circles and squares denote convective heat transfer links and metal temperatures respectively.

![Figure 4: Discretization of a shell-and-tube heat exchanger.](image4)

Figure 5 shows the discretization scheme for the pebble bed reactor. Again the larger circles and squares denote flow paths while the smaller open circles denote convective heat transfer links to the surface of the fuel pebbles (denoted by the smaller open squares). The solid circles and squares denote conductive heat paths inside the pebbles while the semi-solid circles denote radiation and convective heat transfer in the packed pebble bed.

![Figure 5: Discretization of the pebble bed reactor.](image5)

The model can also handle 2-D conductive heat transfer structures connected to flow elements. The discretization of these elements is beyond the scope of this paper.

NUMERICAL APPROACH

The numerical method, which can be described as an implicit pressure correction method (IPCM), has been described elsewhere by the author with reference to single pipe lines [1]. Following is a brief explanation of the method with
reference to unstructured meshes as encountered in generalised networks.

**Governing Equations**

The equations governing the solution of mass flows, pressures and temperatures are the continuity, momentum and energy equations.

Consider a general node i with J branches as shown in Figure 6.

![Figure 6: General node with neighbouring nodes connected through branch elements.](image)

The continuity equation for node i can be expressed as:

\[
\frac{\partial \rho_i}{\partial t} = \sum_{j=1}^{J} \rho_j Q_j s_j + d_i \tag{1}
\]

where \(s_j\) is the ‘sign’ of element j with \(s_j = 1\) if the positive flow direction of element j is from node j to node i and \(s_j = -1\) if the positive flow direction is from node i to node j.

The momentum equation for element j can be written in the following form:

\[
f(p_j, Q_j) + \frac{\rho_j \Delta x}{A} \frac{\partial Q_j}{\partial t} + s_j (p_i - p_j) = 0 \tag{2}
\]

The energy conservation equation for node i is given by:

\[
\frac{\partial (m_i h_i)}{\partial t} - \sum_{j=1}^{J} \frac{\partial p_j}{\partial t} = \sum_{j=1}^{J} \left[ s_j m_j (h_j + g z_j) + \dot{E}_j \right] + \]

\[
d_i (h_{\text{en}} + g z_i) + (h_i + g z_i) \sum_{j=1}^{J} s_j m_j - d_i + \dot{E}_i \tag{3}
\]

**Solution algorithm**

When solving the set of equations (1) to (3) one can either use an explicit method, such as the two-step Lax-Wendroff method or the Method of Characteristics, or an implicit method. The advantages of explicit methods are that they are generally simpler to program and faster than implicit methods per time step. The stability of explicit methods is, however, governed by the \(\Delta x - \Delta t\) relationship, which implies that \(\Delta t\) will be determined by the shortest length increment in the system. This usually results in very small time steps, which makes explicit methods very slow when calculating slow transients or steady-state flows where many time steps are required to cover the time period under interest. Since most transient simulations require the steady-state solution as the initial condition, the speed advantage of explicit methods per time step is cancelled by the fact that many time steps are required to calculate the steady-state condition.

The main advantage of implicit methods is that they are formally unconditionally stable, which means that the restrictive \(\Delta x - \Delta t\) relationship that applies to explicit methods, does not apply to implicit methods. Although accuracy may be lost when using large time steps, most of the transients that need to be analysed in control studies can be classified as slow transients. For slow transients one can usually use time steps much larger than that predicted by the \(\Delta x - \Delta t\) relationship without loss of accuracy. It was therefore decided to follow an implicit approach.

The method used to solve the flow and pressure fields can be classified as a simultaneous node pressure correction method. Following is a brief description of the method. Integration of (1) over a time step \(\Delta t\) yields

\[
\frac{\rho_i - \rho_i^*}{\Delta t} = \alpha \left[ \sum_{j=1}^{J} \rho_j Q_j s_j + d_i \right] +
\]

\[
(1 - \alpha) \left[ \sum_{j=1}^{J} \rho_j Q_j s_j + d_i \right]^o \tag{4}
\]

where terms with the superscript \(o\) are evaluated at the previous time level and terms without a superscript are evaluated at the new time level. \(\alpha\) is a weighing factor between the new and old time levels. When \(\alpha = 1\) the scheme becomes fully implicit and when \(\alpha = 0\) the scheme becomes explicit. When \(\alpha = 0.5\) the scheme is equivalent to the Crank-Nicholson time integration scheme, which is second order accurate in time.

The continuity error at node i can therefore be expressed as:

\[
r_i = \alpha \left[ \sum_{j=1}^{J} \rho_j Q_j s_j + d_i \right] + (1 - \alpha) \left[ \sum_{j=1}^{J} \rho_j Q_j s_j + d_i \right]^o -
\]

\[
\frac{\rho_i - \rho_i^*}{\Delta t} \tag{5}
\]

Integration of (2) over a time step \(\Delta t\) yields:

\[
\alpha \left[ f(p_j, Q_j) + s_j (p_i - p_j) \right] +
\]

\[
(1 - \alpha) \left[ f(p_j, Q_j) + s_j (p_i - p_j) \right]^o +
\]

\[
\rho_j \Delta t \left( \frac{Q_j - Q_j^*}{\Delta t} \right) = 0 \tag{6}
\]

where \(f = f(\rho, Q)\) such as the Darcy-Weisbach equation which expresses the frictional pressure drop in terms of density and volumetric flow rate.
we can write that
\[ r = r_i(p_1, ..., p_n) \quad (7) \]
for all nodes in the network.

According to the Newton-Raphson method when there are \( n \) equations to be satisfied \( r(p_1, ..., p_n) = 0, ..., r_n(p_1, ..., p_n) = 0 \) and \( n \) unknowns \((p_1, ..., p_n)\) to be solved for, the set of improvements \((\Delta x_1, ..., \Delta x_n)\) is the solution of the set of simultaneous linear equations:
\[ r + \sum_{k=1}^{n} \frac{\partial r}{\partial p_k} \Delta p_k = 0 \quad i = 1, 2, ..., n \quad (8) \]

Substitution of (5) into (8) and assuming constant densities we get:
\[ r + \alpha \left( \sum_{j=1}^{n} s_j \rho_j \frac{\partial Q_j}{\partial p_i} - \frac{\partial \rho_i}{\partial \Delta t} \Delta p_i \right) + \alpha \sum_{j=1}^{n} s_j \rho_j \frac{\partial Q_j}{\partial p_i} \Delta p_j = 0 \quad i = 1, 2, ..., n \quad (9) \]

Using (6) to evaluate the \( \partial Q/\partial p \) terms in (9) we get
\[ \frac{\partial Q_j}{\partial p_i} = -\frac{s_j}{\rho_j} \frac{\partial f_j}{\partial p_i} + \rho_j \Delta x_j \frac{\Delta x_j}{\Delta t} \quad (10) \]
and
\[ \frac{\partial Q_j}{\partial p_j} = \frac{s_j}{\rho_j} \frac{\partial f_j}{\partial p_j} + \rho_j \Delta x_j \frac{\Delta x_j}{\Delta t} \quad (11) \]

The term \( \partial Q/\partial p \) in (9) can be evaluated with an equation of state. Substitution of (10) and (11) into (9) leads to an equation which, when solved for the pressure corrections. The pressure corrections are now applied to the pressures at the previous time level to give an updated pressure field at the new time level. Following the updating of pressures, the flow field is corrected using (6). The process is repeated until the flow and pressure field are sufficiently converged at the new time level. Next the temperature field is solved (as will be discussed later) after which the process of alternately solving the flow/pressure fields and temperature fields is repeated until convergence. This gives the solution of flows, pressures and temperatures at the new time level. The time is now progressed to the new level and the whole process repeated. Although the algorithm seems laborious, convergence is quite fast and robust.

Equation (3) can be written in the following form:
\[ \frac{\partial (m_i h_i)}{\partial t} - \frac{\partial \rho_i}{\partial t} = B_i \quad (12) \]
which when integrated over a time step gives
\[ \frac{m_i h_i - m'_i h'_i}{\Delta t} - \frac{\partial \rho_i}{\partial t} = \alpha B_i + (1 - \alpha)B'_i \quad (13) \]

Written in the proper form Eq. (13) gives an equation, which can be solved for the enthalpy field. Using thermodynamic relationships the enthalpy values can be converted to temperatures.

RESULTS

A few examples emanating from a benchmark study will be presented in this section.

Pressure waves in a pipe due to the sudden closure of a valve

The first example is the simulation of pressure pulses due to the sudden closure of a valve at the downstream end of a 20 m, 0.5 m diameter pipe. The fluid is helium. The simulation starts with the steady-state solution in which the inlet pressure is 700 kPa, the inlet temperature is 283 K and the outlet pressure is 680kPa. This gives an outlet Mach number of 0.21 prior to the closing of the valve. A constant friction factor of 0.02 was used while the pipe was divided into 20 increments.

Figure 7 shows the pressure variation at the downstream end of the pipe for the case of adiabatic flow. The results of the following cases are presented: the present method with a time integration factor \( \alpha=1 \), the present method with \( \alpha=0.6 \), and the two-step Lax-Wendroff method [2], which is a second order explicit method. A time step of 0.0009 was used, which is approximately 90 percent of that prescribed by the Courant condition [2].

The two-step Lax-Wendroff (LW) method and present method with \( \alpha=0.6 \) compare very well. The present method with \( \alpha=1 \) exhibits a larger dampening of the pressure wave. This is due to the numerical approximation error of the time derivative term. This error cannot be reduced by refining the grid spacing, as \( \alpha \) has no effect on the accuracy of the spatial derivative term. An obvious method to reduce the numerical approximation error is to decrease the time step. This will, however, slow down execution time. A better approach is therefore to decrease the value of \( \alpha \).

With \( \alpha=1 \) the present method is only first order accurate in time while it is second order accurate when \( \alpha=0.5 \). The method, however, becomes unstable for \( \alpha \)-values close to 0.5 and it was found that a \( \alpha \)-value of 0.6 offers a good compromise between accuracy and stability.

The execution times of the two methods for a 2 second simulation period on a 386 PC were as follows: two-step Lax-Wendroff method, 0.8 s; and the present method, 24 s. This implies that the present method is approximately 30 times slower than the two-step Lax-Wendroff method. It should, however, be kept in mind that the present example represents a fast transient case where the time step had to be made approximately the same as that of the two-step Lax-Wendroff method to achieve the same level of accuracy. However, in the case of slow transients the time step of the implicit method can be made much larger than that of the two-step Lax-Wendroff method without significant loss of accuracy, which makes the implicit method faster than the explicit method in the case of slow transients.
Figure 7: Variation of pressure at the end of a 20 m long, 0.5 m diameter pipe due to the sudden closure of a valve at the downstream end of the pipe for the case of adiabatic flow.

Heat transfer through a reservoir wall due to varying ambient conditions

In this example a cylindrical reservoir of 10 m³, with water flowing through at a constant rate of 5 kg/s at 323.15 K is considered. The reservoir wall consists of three layers with characteristics as shown in Table 1. The outside of the reservoir is exposed to air at 373.15 K.

Starting at steady-state conditions the outside convection temperature is varied with a step function from 273.15 K to 373.15 K for 100-second intervals.

Figure 8 shows the temperatures variation with time for the outside, middle and inside of the reservoir respectively. The IPCM results are compared with an analytical solution (BM). As can be seen the results correlate well.

<table>
<thead>
<tr>
<th>Layer</th>
<th>Thickness [m]</th>
<th>λ [W/m²K]</th>
<th>ρc_p [KJ/m³K]</th>
<th>Outside Area [m²]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.02</td>
<td>20.0</td>
<td>2000</td>
<td>10.1</td>
</tr>
<tr>
<td>2</td>
<td>0.03</td>
<td>30.0</td>
<td>1500</td>
<td>10.25</td>
</tr>
<tr>
<td>3</td>
<td>0.04</td>
<td>40.0</td>
<td>3000</td>
<td>10.45</td>
</tr>
</tbody>
</table>

Table 1: Layer properties

Figure 8: Temperature transient in a reservoir wall due to varying ambient conditions.

Temperature transient in a recuperator

In this example a recuperator with a total heat transfer area of 300 m² and a thermal capacitance of 9 kJ/K is considered. The hydraulic diameter and cross flow area of both conduits are 0.5 m and 0.1963 m² respectively. Both the hot and cold fluids are helium at an inlet pressure of 700 kPa and an outlet pressure of 670 kPa. Initially the total inlet temperature of both streams is the 300 °C. Starting at steady-state conditions the total inlet temperature of the hot stream is stepped to 500 °C at time t=0 s.

Figure 9 shows the outlet temperatures with time of the hot and cold streams for a counter flow arrangement. The results of the IPCM are compared to that of the two-step Lax-Wendroff (LW) method.

The LW method shows a fluctuation in the hot outlet temperature shortly after the beginning of the transient. After about 0.03 s the curve becomes smooth with a very good agreement between the IPCM and the LW method.

Figure 9: Recuperator exit temperatures due to a temperature step at the inlet of the hot stream for the case of counter flow.

Figure 10 shows the outlet temperatures with time of the hot and cold streams for a parallel flow arrangement. Again the agreement between the IPCM and the LW method is very good apart from the temperature fluctuation of the LW method at the start of the transient.

Figure 10: Recuperator exit temperatures due to a temperature step at the inlet of the hot stream for the case of parallel flow.
Load rejection in the PBMR

In this example a load rejection case of the PBMR plant is considered. Figure 11 shows the network representation of the PBMR plant. Initially the plant operates at maximum power and at time $t=0.9$ s the generator power is reduced by 50 percent. The generator speed is controlled by a PID controller that adjusts the bypass flow by opening and closing a control valve.

Figure 12 shows the variation in turbine temperatures while Figure 13 shows the variation in compressor temperatures during the transient.

The example demonstrates the capability of the model to predict transient flows in complex thermal-fluid systems.

![Network presentation of PBMR](image)

**Figure 11:** Network presentation of PBMR.

![Turbine temperatures](image)

**Figure 12:** Turbine temperatures.

**Figure 13:** Compressor temperatures.

CONCLUSION

An implicit finite difference method for the prediction of transient flows in general thermal-fluid networks has been presented in this paper. Although only a few gas flow cases have been presented, the method is suitable for both liquid and gas flows, for both isothermal and non-isothermal flows and for both fast and slow transients. Although the method is slower than explicit methods when applied to fast transients, it is faster than explicit methods in the case of slow transients.

The main advantages of the present method are its average speed over a range of problems including both fast and slow transients, its accuracy and its stability. The method is also very flexible and has been extended to deal with a large variety of non-pipe components such as compressors, turbines, orifices, pumps and heat exchangers.

ACKNOWLEDGMENTS

This work was carried out in association with M-Tech Industrial on contract for PBMR (Pty.) Ltd.

REFERENCES
